REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. REPORT DATE *(DD-MM-YYYY)*	2. REPORT TYPE	3. DATES COVERED *(From - To)*
14-05-2007	Conference Paper (Preprint)	May 2007 – May 2007

4. TITLE AND SUBTITLE

Pulsed Injection Flow Control for Throttling in Supersonic Nozzles – A Computational Fluid Dynamics Design Study (Preprint)

5a. CONTRACT NUMBER

5b. GRANT NUMBER

5c. PROGRAM ELEMENT NUMBER

6. AUTHOR(S)

Dan Baruzzini, Neal D. Domel and Daniel N. Miller

5d. PROJECT NUMBER
DARP052Z

5e. TASK NUMBER

5f. WORK UNIT NUMBER

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)

Lockheed Martin Aeronautics Company
Propulsion Systems and CFD Technologies
Post Office Box 748, Mail Zone 9333
Fort Worth, TX 76101

8. PERFORMING ORGANIZATION REPORT NUMBER

AFRL-PR-ED-TP-2007-276

9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES)

Air Force Research Laboratory (AFMC)
AFRL/PRS
5 Pollux Drive
Edwards AFB CA 93524-7048

10. SPONSOR/MONITOR'S ACRONYM(S)

11. SPONSOR/MONITOR'S NUMBER(S)
AFRL-PR-ED-TP-2007-276

12. DISTRIBUTION / AVAILABILITY STATEMENT

Distribution A: Approved for public release; distribution unlimited (PA # 07197A).

13. SUPPLEMENTARY NOTES

For presentation at the 37th AIAA Fluid Dynamics Conference, Miami, FL, 25-28 Jun 2007. Paper No. AIAA-2007-4215.

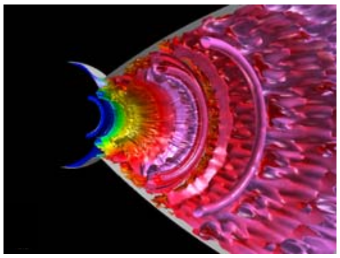
14. ABSTRACT

A vehicle propelled by an engine with a variable geometry nozzle allows the nozzle expansion ratio to vary with altitude and flight condition, thereby optimizing engine performance. Active flow control offers a method of providing the functionality of a variable throat area system without requiring variable geometry. Throttling the mass flow rate through the nozzle throat controls the effective throat area, subsequently controlling the effective expansion ratio of the overall nozzle. This paper presents findings from the Pulsed Injection for Rocket Flow Control Technology (PIRFCT) program, which evaluated potential gains in the overall performance of a rocket using active flow control to optimize nozzle expansion ratio for an Earth to orbit mission. Lockheed Martin Aeronautics Company utilized Computational Fluid Dynamics (CFD) to simulate the rocket nozzle with active flow control. Simulations were performed with steady and pulsed flow control jets which were oriented near the geometric throat and inclined upstream against the primary flow. A low stagnation pressure, steady, tertiary injection stream when combined with a steady, high momentum secondary injector was witnessed to increase throttling performance beyond that of a secondary injector alone. Nozzle discharge coefficient was largely unaffected by changes in pulsation frequency or pulsation duty cycle. Pulsed injection approached, but did not exceed, the throttling performance of a time invariant injector when compared on a equivalent mass flux, momentum flux, and energy flux basis. Simulations incorporating a single injector and large area modulations predicted a 50% area reduction when injecting approximately 18% baseline reference mass flow at Mach 2 conditions. However, the PIRFCT program concluded that secondary injection at the nozzle throat is not a good candidate for this type of throttling/altitude compensation technology for an Earth to orbit mission. This was due to the small portion of its trajectory spent at lower altitudes because of its space access mission. However, potential cadidates for this technology include gas turbines and rockets whose application required the vehicle to stay in the lower atmosphere for a longer duration than a Earth to orbit space access mission.

15. SUBJECT TERMS

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
					1Lt Jannette J. Cohen
a. REPORT	b. ABSTRACT	c. THIS PAGE			19b. TELEPHONE NUMBER *(include area code)*
Unclassified	**Unclassified**	**Unclassified**	SAR	16	N/A

Pulsed Injection Flow Control for Throttling in Supersonic Nozzles– A Computational Fluid Dynamics Design Study (Preprint)



Dan Baruzzini[*], Neal Domel[℘], Daniel N. Miller[∀]
Lockheed Martin Aeronautics Company, Fort Worth, Texas 76101

Abstract

A vehicle propelled by an engine with a variable geometry nozzle allows the nozzle expansion ratio to vary with altitude and flight condition, thereby optimizing engine performance. Active flow control offers a method of providing the functionality of a variable throat area system without requiring variable geometry. Throttling the mass flow rate through the nozzle throat controls the effective throat area, subsequently controlling the effective expansion ratio of the overall nozzle. This paper presents findings from the Pulsed Injection for Rocket Flow Control Technology (PIRFCT) program, which evaluated potential gains in the overall performance of a rocket using active flow control to optimize nozzle expansion ratio for an Earth to orbit mission. Lockheed Martin Aeronautics Company utilized Computational Fluid Dynamics (CFD) to simulate the rocket nozzle with active flow control. Simulations were performed with steady and pulsed flow control jets which were oriented near the geometric throat and inclined upstream against the primary flow. A low stagnation pressure, steady, tertiary injection stream when combined with a steady, high momentum secondary injector was witnessed to increase throttling performance beyond that of a secondary injector alone. Nozzle discharge coefficient was largely unaffected by changes in pulsation frequency or pulsation duty cycle. Pulsed injection approached, but did not exceed, the throttling performance of a time invariant injector when compared on a equivalent mass flux, momentum flux, and energy flux basis. Simulations incorporating a single injector and large area modulations predicted a 50% area reduction when injecting approximately 18% baseline reference mass flow at Mach 2 conditions. However, the PIRFCT program concluded that secondary injection at the nozzle throat is not a good candidate for this type of throttling/altitude compensation technology for an Earth to orbit mission. This was due to the small portion of its trajectory spent at lower altitudes because of its space access mission. However, potential cadidates for this technology include gas turbines and rockets whose application required the vehicle to stay in the lower atmosphere for a longer duration than a Earth to orbit space access mission.

---

[*] Aeronautical Engineer, Propulsion Systems.

[℘] Aeronautical Engineer Senior Staff, Propulsion Integration, AIAA Lifetime Senior Member.

[∀] Lockheed Martin Fellow, Vehicle Sciences & Systems, AIAA Associate Fellow.

Nomenclature

A	Area	$\dot{m}H$	Energy flux
a	Slope of linear correlation	p	Static pressure
C_d	Discharge coefficient	P_t	Stagnation pressure
C_{d_0}	Baseline C_d (no injection)	St	Strouhal number
d	Diameter	T	Static Temperature
$\tilde{f}$	Frequency of pulsing	T_t	Stagnation Temperature
f	Injected momentum flux	χ	Independent variable
F	Primary momentum flux	ρ	Density
H	Stagnation Enthalpy	V	Velocity
I_{sp}	Vacuum specific impulse	γ	Ratio of specific heats
L	Length		
M	Mach number		
$\dot{m}$	Mass flux		

Subscripts

inj	Injected flow quantity	pri	Primary flow quantity
0	Baseline flow quantity (no injection)		

1.0 Introduction

1.1 Problem Declaration

The Air Force Research Laboratory (AFRL) tasked a team of three companies to perform a multidisciplinary analysis of the potential benefits of PIRFCT for an Earth-to-orbit rocket. Lockheed Martin Aeronautics Company had the role of simulating the nozzle with CFD for steady and pulsed injection, and providing injection requirements to achieve a desired amount of throttling for a geometrically fixed throat. The CFD results were used to assess rocket engine performance with a constricted throat. The vehicle's performance was assessed for Earth to orbit trajectories. This paper focuses on the CFD simulations employing active flow control.

1.2 Problem Description

Pulsed injection has been proposed as an alternative to steady injection for various applications involving propulsion and active flow control.[1-4] Although Gunter and Farenholz[5] considered rocket engine throat area reduction utilizing steady injection of a secondary fluid, the current study investigated whether additional throttling gains could be realized by employing pulsed injection for throat area control. The geometric parameters comprising typical simplified rocket nozzle contours possessed expansion ratios of 10 and 19.

1.3 CFD Solver

The CFD solver utilized throughout the study is an internally developed CFD solver given the name of FALCON. FALCON is a compressible, implicit, finite volume solver with options for either a Smagorinsky[6] based Large Eddy Simulation (LES), or 2 equation Reynolds Averaged Navier-Stokes (RANS) turbulence model. FALCON uses the convective upwinded flux difference splitting method of Roe[7] and is up to third order accurate (maximum) in space, second order accurate in time. FALCON's viscous fluxes are centrally difference and incorporate optional MINMOD (Roe[7]) or SUPERBEE (Sweby[8]) flux limiters. Falcon operates in parallel on a structured mesh with a multi-block topology.

1.4 Compressible Pulsing Boundary Condition

A compressible pulsed boundary condition was developed specifically for this analytical investigation, the details of which are described in a companion paper.[9] The boundary condition resembles a modified square wave with rise and fall behavior invoking 1D unsteady characteristic relationships. This modification mimics the behavior of a diverging nozzle which accelerates the injected pulsed flow to supersonic speeds. Additionally, the compressible pulsed boundary condition allows for a user-specified duty cycle and injection angle. A graphical illustration of the pulsed compressible boundary condition's behavior is given in ***Figure 1***.

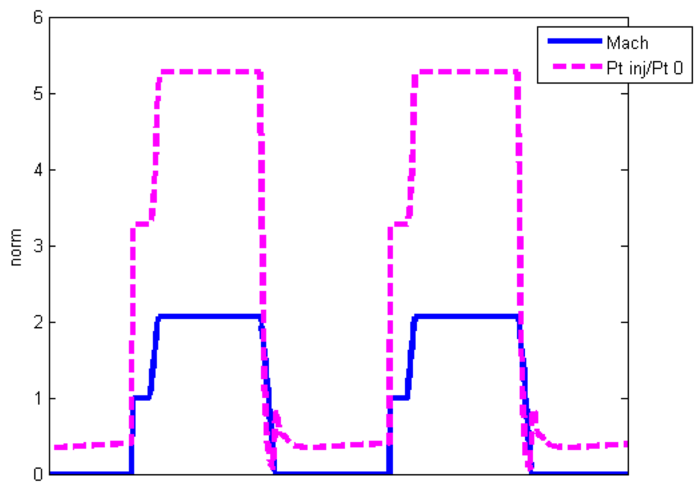


Figure 1. *Mach number and stagnation pressure response across the injection slot over 2 complete cycles*

1.5 CFD Modeling Details and Assumptions

Because the compressible pulsed boundary condition mimics a diverging nozzle for supersonic injection, the pulsed injector was modeled with this boundary condition applied to boundary faces which are flush with the surface of the nozzle. The interior geometry of the injector jet was not included in the simulations. This simplification allowed grids to be reused with little or no changes for the various injection conditions.

The CFD simulations were performed without modeling chemistry or other high temperature real gas effects. Although such effects are relevant when numerically

assessing rocket engine performance, the fundamental fluid mechanics of steady and unsteady active flow control techniques do not require consideration of these effects. This evaluation was conducted by simulating air at a constant specific heat ratio of 1.13, which was representative of the rocket engine's primary core flow. The injected gas was also modeled with the identical value for γ (i.e., single species perfect gas simulation). The pressures and temperatures of the injected fluid were set to deliver a prescribed Mach number and total velocity for a given injected state.

Although the physical nozzle geometry was axi-symmetric, the LES turbulence model appropriately requires a 3 dimensional simulation in order to capture the 3 dimensional turbulent fluctuations in the flow. Therefore, the CFD simulations were performed on periodic (angular) sectors of the rocket geometry ranging from 20 degrees to 45 degrees. Solutions on sectors smaller than 20 degrees were discarded because they appeared to suffer from an inability to capture the largest turbulent structures. Mesh densities were typically $2x10^6$ to $3x10^6$ hexahedral elements and frequently employed a non-point to point interface approximately 100 injector slots lengths downstream of the injection location, which was approximately half the length of the divergent section of the rocket nozzle. Subsequent fully continuous, non-interrupted, hexahedral meshes later confirmed a well-positioned non-point to point interface assumption did not affect the accuracy of the final solution.

2.0 Area Reduction Correlation

2.1 Steady Injection Versus Pulsed Injection

In order to determine the optimum throttling performance of the various flow control schemes, a fair metric of comparison was needed to evaluate the throttling performance of both steady injection and pulsed injection. For this study, the basis of comparison was the combination of the three time averaged fluxes being injected into the main flow. The pulsed mass flux, momentum flux, and energy flux may be measured and appropriately time-averaged in order to obtain "equivalent steady" fluxes over a given injection cycle. When considering steady injection, the steady mass flux, momentum flux, and energy flux are at all times equal to their respective instantaneous values. The resulting mass flux, momentum flux, and energy flux entering the system can be considered the inputs or alternately, the cost required to achieve a given level of throttling performance. The resulting output response to the injected mass flux, momentum flux, and energy flux can be described by a typical discharge coefficient. The equations for the discharge coefficient and injected fluxes are given below in ***Equations 1 - 4***.

$$C_d \equiv \frac{\dot{m}_{primary}}{\dot{m}_{ideal}} \qquad \textit{(Equ. 1)}$$

$$\dot{m} \equiv \overline{A(\rho V)} \qquad \textit{(Equ. 2)}$$

$$f \equiv \overline{A\left(\rho V^2 + p - p_{throat}\right)} \qquad \textit{(Equ. 3)}$$

$$\dot{m}H \equiv \overline{A(\rho V)\left(c_p T + \tfrac{1}{2}V^2\right)} \qquad \textit{(Equ. 4)}$$

It can be seen that each flux scales linearly with area. Therefore the area of the injector jet is the key for linking an equivalent steady case to a case pulsing with a square wave at a known duty cycle. That is, a pulsing case with a 50% duty cycle has twice the jet area as its equivalent steady case. Similarly, a case with a 25% duty cycle would have four times the area of its equivalent steady state counterpart.

Traditionally, comparisons between pulsed and steady injection have been performed with equal jet areas, and the injection velocity or pressure is adjusted to preserve an equivalent mass flow. However, from ***Equations 1-4*** it is apparent that neither velocity nor pressure can be appropriately isolated for consistent scaling of all fluxes. As a result, attempts to increase the "pulse-on" pressure or velocity, in order to match a steady mass flux, will produce an inconsistent increase in the momentum and energy fluxes, and subsequently predicts an increased level of aerodynamic blockage. Consequently, to ensure a fair comparison during the entire computational investigation, the injector area was resized and a new mesh was created when equivocating steady versus pulsed injection, rather than increasing injection velocity or pressure.

2.2 Throttling Correlation

A typical nozzle throat area reduction correlation approach commonly describes the discharge coefficient as a second or third order polynomial, dependant on either mass flux or momentum flux.[10,11] After a number of (steady and unsteady) injection simulations had been conducted under various injection conditions during the program, it was observed that a general correlation was dependent upon all three fluxes (mass, momentum and energy). A single independent variable, involving all three fluxes, was derived which sufficiently captures all of the CFD simulation results given below in ***Equation 5***.

$$\chi = \sqrt{\frac{f_{inj}}{F_{pri}}\sqrt{\frac{\dot{m}_{inj}}{\dot{m}_{pri}}\frac{(\dot{m}H)_{inj}}{(\dot{m}H)_{pri}}}} \qquad \textbf{\textit{(Equ. 5)}}$$

An in depth description of the correlation is described in a companion paper[9] but the simulation results across the entire program are given below in terms of ***Equation 5*** in a linear format in ***Figure 2***.

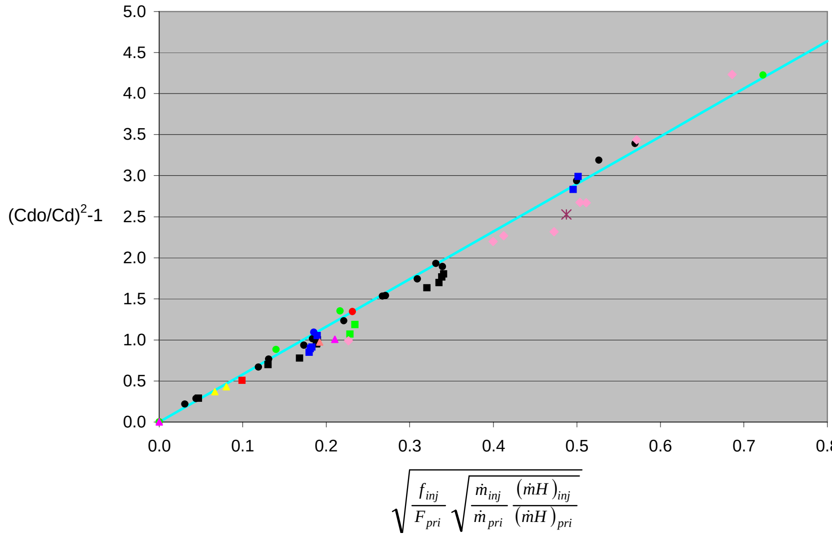


Figure 2. *Linear throttling correlation relating nozzle discharge coefficient as a function of injected mass flux, momentum flux, and energy flux*

The elegance of this particular correlation is that it allows for an impartial metric of comparison when judging CFD simulations with widely varying secondary injection

mass flow rates, pressures, temperatures, or velocities. Additionally, note that from ***Figure 2*** and ***Equation 6***

$$c_d = \frac{c_{d0}}{\sqrt{1+ax}} \qquad \textbf{\textit{(Equ. 6)}}$$

that the linear correlation slope is completely defined by the constants "a" and Cdo. Cdo is the baseline value of discharge coefficient (no injection), and is a function of geometry and primary flow conditions. The constant "a" is a function of these, as well as injection location and orientation (but not injection stream conditions). From ***Figure 2*** it stands to reason that the ultimate nozzle throat area reduction is achieved when "a" approaches infinity (infinite throttling at no cost) and conversely, the poorest nozzle throat area reduction is cited when "a" is equal to zero (no throttling at infinite cost). After conducting numerous rocket nozzle CFD simulations employing various injection rates, temperatures, velocities, and even multiple injection streams, the correlation constant "a" was determined to be approximately 5.8. This constant turned out to be fairly insensitive to the range of nozzle contour variations in this study. However, small changes in "a" were observed as various injection locations and orientations were modeled. Subsequently, a CFD simulation employing a particular injection scheme which resulted in a higher value of "a" was considered an improvement. Conversely, a CFD simulation with a lower value of "a" was considered a detriment when seeking an optimal throttling (injector) configuration.

3.0 CFD Simulations

3.1 Representative Time Invariant CFD Simulation

A nominal, time invariant, RANS CFD simulation employing 8% mass injection at an injected Mach number of 2 and inclined 45 degrees with respect to the upstream flow direction is depicted in ***Figure 3***.

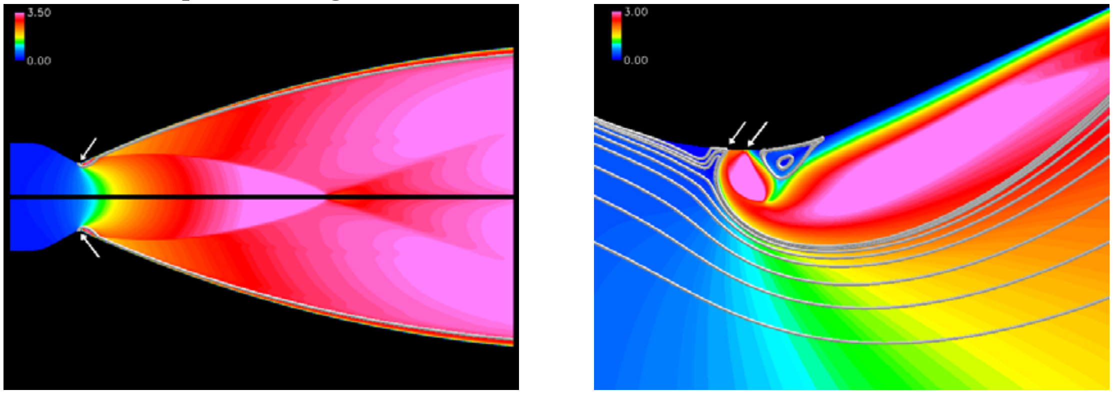


Figure 3. *Mach number distributions across the entire rocket nozzle (left) and in the immediate vicinity of the slot injector (right) for a representative steady state simulation*

The arrows in ***Figure 3*** indicate the leading and trailing edge of the location of the injector while the streamlines highlight the trajectories of the primary and injected fluid streams. Also visible in ***Figure 3*** is an oblique shock required to turn the supersonic flow

in the divergent portion of the rocket nozzle. Lastly, ***Figure 3*** illustrates a persistent low pressure, recirculation region immediately downstream of the injected flow. This separation region is a common characteristic associated with a time invariant RANS CFD simulation employing a single injection stream.

3.2 Pulsed LES CFD Simulations – Variation on Strouhal Number

Strouhal number was defined throughout the analytical investigation as

$$St = \frac{\tilde{f}L}{V}$$

with the length scale "L" denoting the physical injector slot width and V denoting the total velocity of the injected fluid. Numerous CFD simulations were conducted varying injector Strouhal number from 0.02 to 0.47 at a consistent 50% duty cycle. The prevailing trend indicated that the CFD simulations employing a low Strouhal number resulted in lower throttling performance than an equivalent simulation employing steady-state injection. One may expect that for exceptionally low injection Strouhal values, the injection-off phase of the simulation would dominate the overall solution and throttling performance would therefore likely suffer. Additionally, behavior of a typical low Strouhal injection value tended to violently surge the primary flow and was only exacerbated when the magnitude of the injected mass flow rate was increased. ***Figure 4*** highlights the vorticity scalar contours of the discrete pulses at a sub par Strouhal injection value of 0.02 while injecting 10% of the baseline reference mass flow at an injected Mach number of 2.0 and injector inclination of 45 degrees with respect to the upstream flow direction.

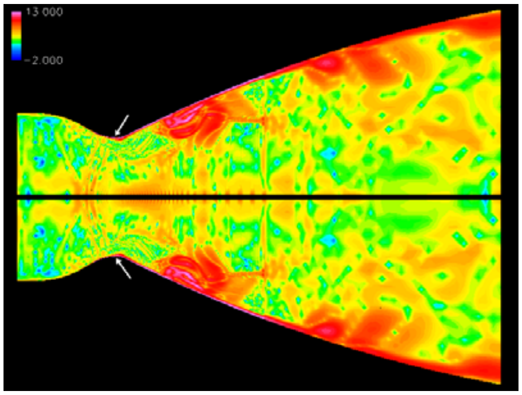


Figure 4. *Magnitude vorticity distributions across the entire rocket nozzle at a sub par Strouhal number of 0.02*

As the injection Strouhal number was increased, the unsteady pulsed solution approached the time invariant solution. A notable flow field characteristic of high Strouhal simulations is the quasi-stable standing recirculation region immediately downstream of the injector. Recall that this feature is characteristic of steady injection simulations. The high Strouhal simulations also mimic the steady solutions in the oblique shock position in the divergent portion of rocket nozzle. ***Figure 5*** illustrates the flow field behavior resulting from a Strouhal injection value of 0.47 while prescribing

approximately 15% of the baseline reference mass flow at an injected Mach number of 1.7 and injector inclination of 45 degrees with respect to the upstream flow direction. Observations from the higher Strouhal number simulations indicate that the blockage achieved by pulsed injection approaches, but does not exceed that of an equivalent simulation employing steady injection. However, the comparison with the equivalent steady injection result improves as the Strouhal number increases.

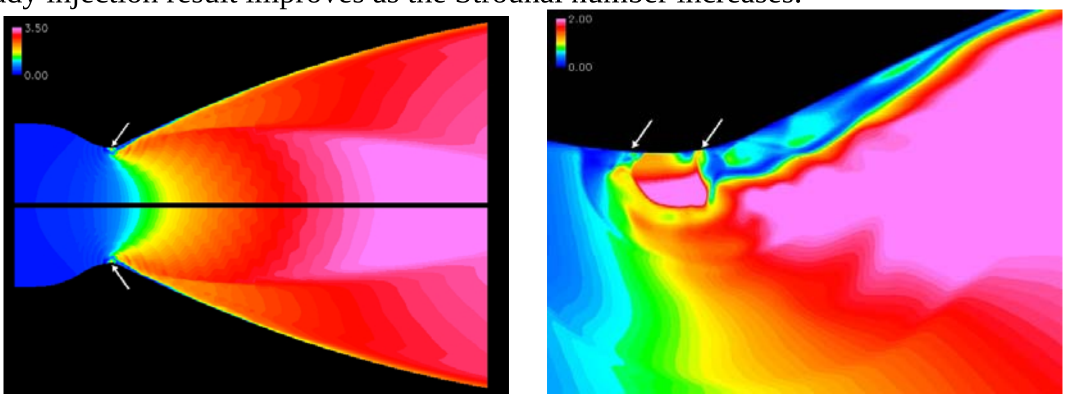


Figure 5. *Mach number distributions across the entire rocket nozzle (left) and in the immediate vicinity of the slot injector (right) for a relatively high injection Strouhal number of 0.47*

3.3 Pulsed LES CFD Simulations – Variation on Duty Cycle

The majority of the CFD simulations were conducted with a 50% duty cycle, but a brief excursion was undertaken to examine how the introduced blockage is affected by varying injector duty cycle. Recall that to maintain an equivalent mass flux, momentum flux, and energy flux between pulsed and time invariant injection, the injector area must be adjusted inversely proportional to the injection duty cycle. For a 50% duty cycle, the pulsed injection area must be identically twice the steady injector area hence, a 25% duty cycle requires 4 times the injection area over the steady injector area. Considering the high degree of curvature at the entrance to the divergent section of the nozzle, duty cycles of 25% or lower required an injector slot width that extended into both the convergent section of the combustor, and more detrimentally, into the rapidly expanding divergent portion of the nozzle. For this reason it is not entirely clear whether duty cycles below 50% exhibited lower gains in throat area reduction strictly due to a reduced duty cycle.

However, an increased duty cycle equivalent to 70% (approximately 1.4 times the equivalent steady injector area) exhibited the identical trend witnessed when increasing injector Strouhal number: as the injector duty cycle was increased, discharge coefficient would decrease. This behavior can potentially be attributed to a pulsed injection simulation with a high duty cycle approaching its time invariant equivalent counterpart. Additionally noteworthy is the consistent position of the oblique shock in the divergent portion of the nozzle in addition to the standing recirculation region down stream of the injector, which are characteristic of a time invariant CFD simulation. ***Figure 6*** shows the flow field behavior resulting from a moderate Strouhal injection value of 0.24 while prescribing approximately 15% of the baseline reference mass flow at an injected Mach number of 1.7 and injector inclination of 45 degrees with respect to the upstream flow direction.

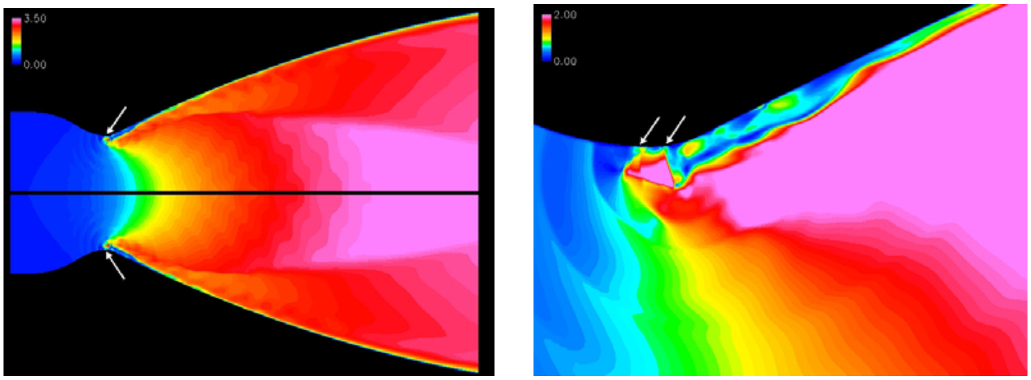


Figure 6. *Mach number distributions across the entire rocket nozzle (left) and in the immediate vicinity of the slot injector (right) for an increased injector duty cycle of 70%*

3.4 Pulsed LES CFD Simulations – Variation on Injection Angle

Multiple CFD simulations were conducted comparing 30 degree injection relative to the upstream flow direction to 45 degree injection in a consistent orientation. Secondary injection to a 90 degree (normal) orientation was not explored due to the prior findings of Miller.[12] Maintaining an equivalent injection Mach number, Strouhal number, duty cycle, and mass flow rate, a slight improvement in nozzle area reduction was observed when injecting the secondary fluid at 30 degrees compared to that of a 45 degree injection scheme. However, the throttling improvement was typically small (on the order of a delta C_d ranging from 0.01 to 0.015). From the vorticity scalar contours given in ***Figure 7*** it is apparent that the flow field in the vicinity of the injectors appear strikingly similar despite a slight reduction in perceived radial penetration. Note that both injection strategies depicted in ***Figure 7*** employ Mach 2.0 injection conditions at a Strouhal number of 0.20 and introduce approximately 10% of the baseline reference mass flow with a 50% duty cycle.

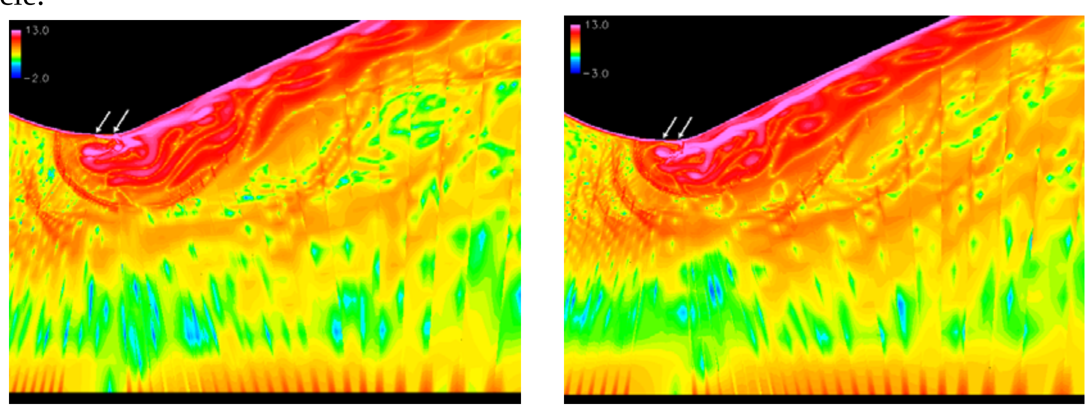


Figure 7. *Magnitude vorticity distributions across a portion of the rocket nozzle in the vicinity of the slot injector at an injection angle inclined 30 degrees (left) and 45 degrees (right) to the upstream flow direction*

3.5 Pulsed LES CFD Simulations – Attempts to Increase Entrainment

During the analytical investigation, an attempt was made to increase throat area reduction by augmenting injector entrainment through various simultaneous combinations of pulsed and steady injectors. One configuration examined twin injectors,

separated by approximately 2 injector slot widths, pulsing in phase at Mach 2.0 with individual injector Strouhal numbers of 0.05, individually introducing approximately 5% of the baseline reference mass flow and employing a typical 50% duty cycle. While intutively one may expect the upstream injector to augment the radial penetration of the downstream injector, the opposite trend was witnessed. A possible explanation for the counter intuative flow field behavior is the trailing injector's proximity to the rapidly expanding flow in the highly curved region of the divergent nozzle. The rearward injector may have been potentially overwhelmed by the strong outward radial velocity imparted by the (locally) severe divergent nozzle contour and therefore, the upstream injector was operating as if the downstream injector were abscent. Consequently, a large portion of the overall mass, momentum and energy flux was unrealized and this dual pulsed injector simulation displayed the poorest throttling performance of the overall investigation. Additionally, increased injector entrainment was not observed as a result of the separation in distance between the dual pulsed injectors. A scalar contour of vorticity created by the dual pulsed jets is highlighted in ***Figure 8***.

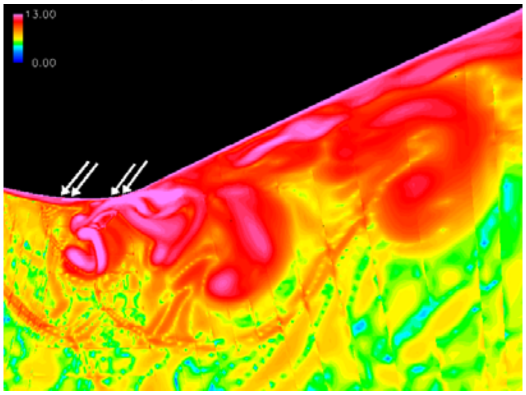


Figure 8. *Magnitude vorticity distribution in the immediate vicinity of the slot injector for dual injector configuration pulsing in phase*

An additional alternate approach attempted to augment injector throttling performance by tightly coupling a steadily flowing Mach 1 jet between 2 identical pulsed Mach 2 jets. The interaction between the 3 injectors made for a fascinating display of fluid physics, but similar to the dual pulsed injector simulation, the overall throttling performance was well below that of the equivalent steady injection simulation.

A final alternate injection approach was to augment injector entrainment by employing a single pulsed injector with a high injection mass flow rate coupled with a low injection frequency. An injector mass flow rate of 20% relative to an unthrottled primary core stream was introduced at Mach 2.0, through a slot angled upstream to the primary flow at 30 degrees at a injection Strouhal number of 0.10 and 50% duty cycle. In general, previous Mach 2.0 injection simulations employed injected mass flow rates ranging from 5% to 10% relative to an unthrottled primary core stream. From ***Figure 9*** the radial penetration of the jet is immediately noticable in the vorticity scalar contours.

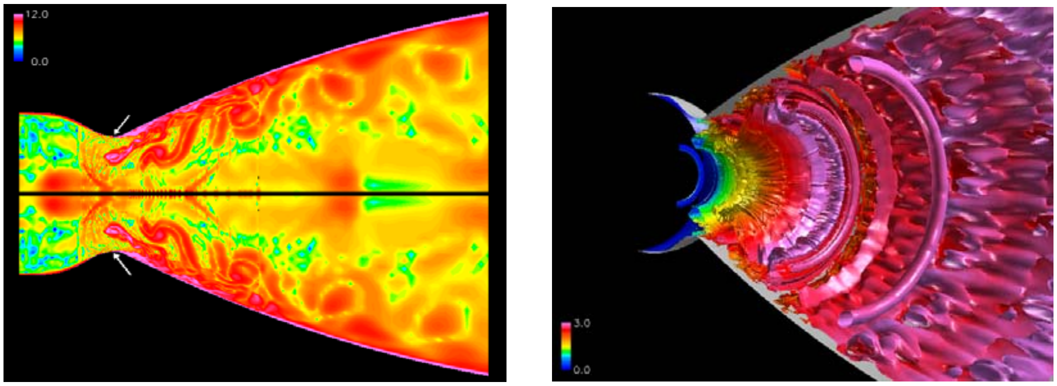


Figure 9. Magnitude vorticity distribution across the entire rocket nozzle (left) and iso-surfaces of constant vorticity scalar colored by Mach number (right) in an aft looking forward orientation

The large scale, 3 dimensional turbulent structures issuing forth from the injection slot are clearly visible in ***Figure 9*** as the energy continually cascades to the intermediate and smaller scales. Also clearly visible in ***Figure 9*** are the nearly perfect, 3 dimensional vortical rings.

A cause for concern predicted by this simulation was the violent disturbances introduced when coupling a high injection mass flow rate at a low frequency. The CFD simulation predicts the presence of strong shock waves due to the relatively long time interval between injection pulses. These shocks had ample time to reflect and impinge on the rocket nozzle walls, introducing localized regions of excessive pressures likely to cause nozzle structural damage.

3.6 Injection of Tertiary Stream

Miller[13] explored the concept of "assisted reinjection" in a gas turbine engine where a desired portion of the higher static pressure, primary core flow, is redirected to a low static pressure region immediately downstream of a secondary injector. Although the application of the reinjector may be a reasonable implementation in a gas turbine application, the redirection of rocket combustor products, in excess of 6000 degrees Rankine, is impractical. However, replacing the reinjector with an additional injector (a tertiary stream of fluid) may be feasible if the fuel and oxidizer comprising the tertiary stream were combusted immediately before the injection.

Numerous steady-state RANS CFD simulations were conducted to investigate the reduction in the discharge coefficient (increased throttling performance) with the addition of a tertiary injection stream. The throttling performance of simulations possessing a tertiary injection stream varied such that some configurations were superior while others were inferior to a single (secondary) injector as referenced in ***Figure 10***.

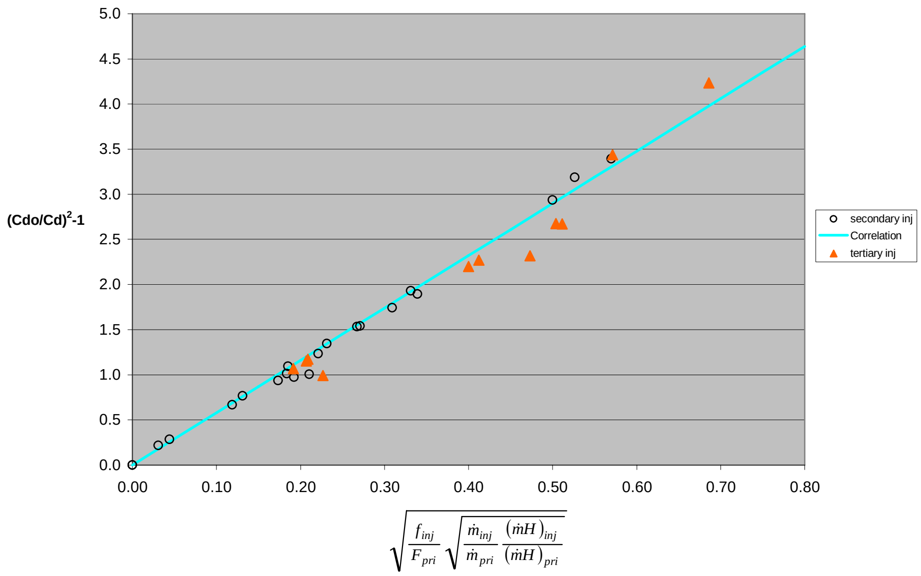


Figure 10. *Linear throttling correlation relating nozzle discharge coefficient as a function of injected mass flux, momentum flux, and energy flux secondary and tertiary injection streams - Secondary injection streams are represented by black symbol coloration; Tertiary injection streams are represented by orange coloration*

A dominant factor regarding throttling performance was the location of the combined secondary and tertiary injection streams relative to the geometric throat. The optimal configuration (as witnessed) with respect to a reduced discharge coefficient at a minimized mass, momentum, and energy flux occurred with the downstream edge of the aft-most injector slot positioned slightly forward of the nozzle geometric throat, graphically illustrated in ***Figure 11***.

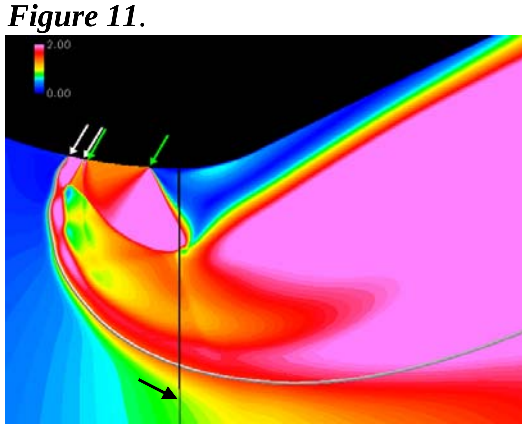


Figure 11. *Mach number distribution in the immediate vicinity of the slot injector detailing secondary and tertiary injection streams; the indicated black line highlights the location of the geometric throat*

4.0 System Implications

The correctly positioned secondary / tertiary injecton stream combination was predicted through CFD to have the lowest discharge coefficient relative to the prescribed input mass, momentum, and energy flux. However, the implications of adding a tertiary

injection stream warrant additional discussion. Consider a low thrust liquid methane oxygen rocket engine with a throat total pressure of 600 psia and total temperature of 6200 degrees Rankine. Due to the high temperatures and chemical composition of the combusted fuel and oxidizer, the resulting ratio of specific heats will be approximately 1.13 assuming an equilibrum based calculation. Isentropic relations would then predict that a a total pressure ratio of 6 (or 3600 psia) is required to deliver an injected stream at Mach 2 assuming injector thermodynamic properties are similar to that of the primary flow core (utilization of a common propellant). If a different rocket engine is examined at a similar throat total temperature but a total pressure of 1500 psia, the required total pressure to drive a secondarily injected Mach 2 jet is now approximately 8000 psia. Rocket hardware necessary to supply such a pressure will probably be both heavy and costly, potentially negating any performance gains realized by a secondary injection at the throat.

In contrast, the tertiary injection stream only requires a plenum total pressure ratio of unity relative to the core flow implying that the hardware and propellants already present on the rocket can be utilized. However, a complex relationship exists between the secondary and tertiary injection streams. The tertiary injection stream tends to reinforce the secondary injector which in turn, depresses the static pressure in the location of the aft injector, which tends to increase the tertiary injection stream’s velocity and mass flow rate. Due to the complex coupled relationship it is difficult to predict a priori the resultant net discharge coefficient of the combined injector scheme. The underlying implication is that a vehicle level system trade study would be necessary in order to determine the optimal design of coupled secondary and tertiary fluid paths.

Although originally not part of the CFD throttling investigation, rocket nozzle ideal specific thrust was calculated for a representative portion of the overall CFD case suite involving an exclusive secondary injector. The trends suggested a high injection mass flow rate with lowered injection momentum is efficient when attempting to optimize throttling but reduces the ideal specific impulse at the rocket nozzle exit. This is believed to be attributed to an overall dilution in the net momentum and energy of the resulting mixed primary and secondary fluids.

5.0 Conclusions

Approximately 50 steady and unsteady CFD simulations were conducted on multiple rocket nozzle contours to investigate active flow control techniques needed to maintain a perfectly expanded rocket nozzle. The steady-state simulations employed a RANS turbulence model, while all pulsed simulations utilized a LES turbulence model. The parametrics within the scope of the analytical investigation were steady injection versus pulsed injection, injection angle, injection location relative to the geometric throat, pulsation Strouhal number, pulsation duty cycle, and simultaneous combinatioins of steady and pulsed injectors. Throttling performance is witnessed to improve when the injector is located slightly forward of the geometric throat. Additionally, injection at 30 degrees in the upstream direction delivered performance slightly superior to that of 45 degree injection. A low stagnation pressure (tertiary) injection stream is observed to increase throttling performance by exploiting the low static pressure recirculation region

created by an upstream (secondary) injector. Nozzle discharge coefficient is largely unaffected by pulsed injector frequency (Strouhal number) unless the injection frequency is exceptionally low. Very low pulsed injection frequencies tended to cause potentially severe fluctuations in the nozzle core mass flow. Very high injection frequencies approach the time invariant result of a quasi-steady, standing recirculation region downstream of the point of injection. Throughout the analytical investigation, pulsed injection approached, but did not exceed, the throttling performance of a time invariant injector of equivalent mass flux, momentum flux, and energy flux. Overall vehicle perfromance analysis conducted external to Lockheed Martin indicated that a Earth to orbit rocket trajectory was not well suited to receive this particular application of active flow control treatment. Subsequently, a follow on program with an alternate application is in progress at the time of this writing.

Acknowledgements

This analytical study was a DARPA funded program managed by AFRL. The author would like to explicitly thank Lt. Col. David Lucia of DARPA for his vision of this program and the staff members at AFRL for their support. The author would also like to thank the involved parties representing WASK Engineering and Advatech Pacific whose talent and dedication made this work possible.

References

- Bender, E., Miller, D.N., Smith, B., Yagel, P., Vermeulen., P., Walker, S., 2000, “Simulation of Pulsed Injection in a Crossflow Using 3-D Unsteady CFD,” AIAA No. 2000-2318.
- Linden, P.F., Turner, J.S., 2001, “The formation of ‘optimal’ vortex rings, and the efficiency of propulsion devices.” Journal of Fluid Mechanics 427, 61-72.
- Tillman, C. P., “Enhancement of Transonic Airfoli Performance Using Pulsed Jets for Separation Control ,” AIAA No. 2001-0731.
- McManus, K., Magill, J., “Separation Control on Incompressible and Compressible Flows Using Pulsed Jets,” AIAA No. 96-1948.
- Gunter, F.L. and Farenholz, F.E., 1961, “Final Report on a Study of Rocket Thrust Control by Gas Injection,” Massachusetts Institite of Technolog, Naval Supersonic Laboratory, Technical Report 448.
- Smagorinsky, J., 1963, “General Circulation Experiments with the Primitive Equations,” Mon. Weather Rev., Vol. 91, pp. 99-164.
- Roe, P.L., 1981, “Approximate Riemann Solvers,” J. of Comp. Physics, Vol. 43, pp. 357-372.
- Roe, P.L., 1981, “Some Contributions to the Modeling of Discontinuous Flows,” Proceedings of the SIAM/SMS Seminar.
- Sweby, P.K., 1984, “High Resolution Schemes Using Flux Limiters for Hyperbolic Conservation Laws,” SIAM J. Numer. Anal., Vol. 21, pp. 995-1101.
- Domel, N.D., Baruzzini, D., Miller, D.N., 2007, “Pulsed Injection for Rocket Flow Control Technology (PIRFCT) – A Computational Fluid Dynamics Analysis,” AIAA No. 2007-4214.

- Miller, D.N., Yagel, P., Bender, E., Smith, B., Vermeulen., P., 2001, "A Computational Investigation of Pulsed Injection Into a Confined, Expanding Crossflow," AIAA No. 2001-3026.
- Catt, J.A., Miller, D. N., 1995, "A Static Investigation of Fixed-Geometry Nozzles Using Fluidic Injection for Throat Area Control," AIAA No. 95-2604.
- Catt, J., Miller, D.N., Walker, S., 1997, "Extending Flow Control of Fixed Nozzles: A Static Investigation of Assisted Reinjection," FEDSM97-3230.
- Miller, D.N., Catt, J., Walker, S., 1997, "Extending Flow Control of Fixed Nozzles Through Systematic Design: Introducing Assisted Reinjection," FEDSM97-3680.